Space Travel

Computational Fluid Dynamics Simulations of Ares I Launch Vehicle

Using computational fluid dynamics (CFD), a team of NASA researchers is estimating aerodynamic effects on the new Ares I Crew Launch Vehicle (CLV) during the ascent phase of the mission. The main goal of the work is to generate a database of aerodynamic estimates, which, along with experimental data, can be used to make intelligent design decisions for the CLV.

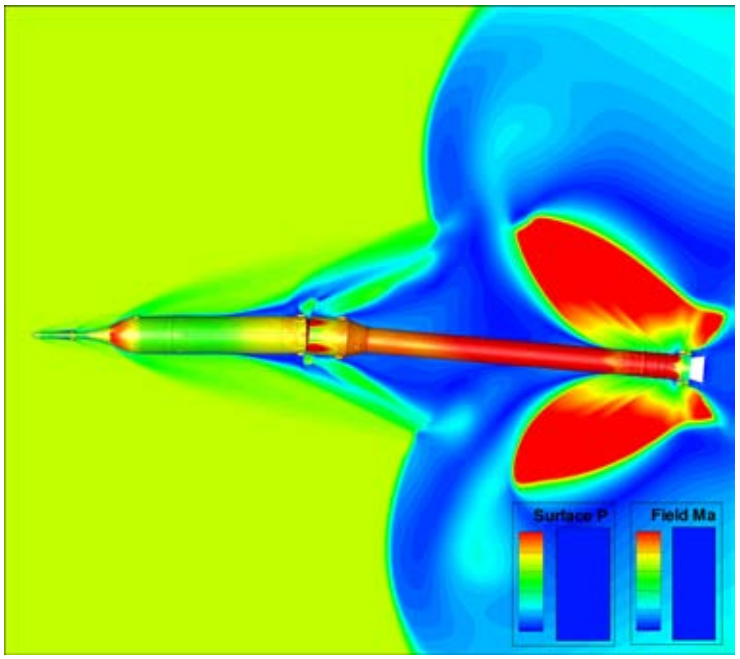
- Specifically, the project involved generating meshes (on which flowfield solutions can be generated), running/monitoring simulations, and post-processing simulation results

- The CFD Navier-Stokes flow solver, OVERFLOW, is being used to simulate the flowfield around Ares I during first stage separation

- The image seen here reveals Mach contours and surface pressures of the resulting flowfield

This work is being done in direct support of NASA's Constellation Program. Critical design decisions for Ares I will be made using the simulation data collected from the team's simulations. The Ares I CLV will be used to carry astronauts into low-earth orbit.

A problem of this scale could not be resolved on a conventional cluster. Most of the simulations consist of about 200 million grid points—each solution file is about 8 gigabytes in size—and requires more than 500 processors per case. The speed and processor availability (about 600 teraflops and 51,000 processor-cores, respectively) of the Pleiades supercomputer at the NASA Advanced Supercomputing facility allows the team to obtain solutions with feasible turnaround time.



The Boost Deceleration Motors (BDMs) provide the thrust necessary to safely separate the first stage of Ares I, while the Ullage Settling Motors (USMs) aid in separation in addition to settling the liquid propellants to the bottom of the J-2X tanks. (Jeff Onufer, NASA)

James Kless, NASA Ames Research Center
James.Kless-1@nasa.gov